ISSN: 2349-0845

International Journal of Research in Science & Technology (IJRST)
Volume 1, Issue 2, April 2014:Page No. 4 - 6
www.ijrst.weebly.com

Design and Optimization of piston using finite element analysis

Nikhil R. Chaple[1], Dr. T.R. Deshmukh[2],

2nd year M.E. CAD/CAM, Department of Mechanical Engineering, Prof. Ram Meghe Institute of Technology and Research Badnera-Amravati.(M.S.) India,nikhilchaple@rediffmail.com[1]

Professor, Department of Mechanical Engineering, Prof. Ram Meghe Institute of Technology and Research Badnera-Amravati. (M.S.) India, [2]

Abstract: This work describes the stress distribution of the piston by using finite element method (FEM). FEM is performed by using computer aided engineering (CAE) software. The main objective of this project is to investigate and analyze the stress distribution of piston at the actual engine condition during combustion process. The parameter used for the simulation is operating gas pressure and material properties of piston. The report describes the mesh optimization by using FEM technique to predict the higher stress and critical region on the component. Aluminium is selected as piston material. It is important to locate the critical area of concentrated stress for appropriate modification. Computer aided design (CAD) software Catia is used to model the piston, and analysis is performed by using ANSYS.

Keywords: Computer CAD, CAE, Optimization, FEA.

I. INTRODUCTION

Automobile components are in great demand these days because of increased use of automobiles. The increased demand is due to improved performance and reduced cost of these components. [2] In internal combustion engine, piston is one of the important components. It reciprocates within the cylinder bore by force produced during the combustion process.

The two main requirements of the piston are as follows: [6]
1- It should contain all the fluids above and below the piston assembly during the cycle.
2- It should transfer the work done during combustion process to the connecting rod with minimal mechanical and thermodynamic losses.

Piston must have some desirable characteristic [6]

- It should be silent in operation both during warming up and the normal running.
- The design should be such that the seizure does not occur.
- It should offer sufficient resistance to corrosion due to some product of combustion.
- It should have the shortest possible length so as to decrease overall engine size.
- It should be lighter in weight.
- It must have a long life.

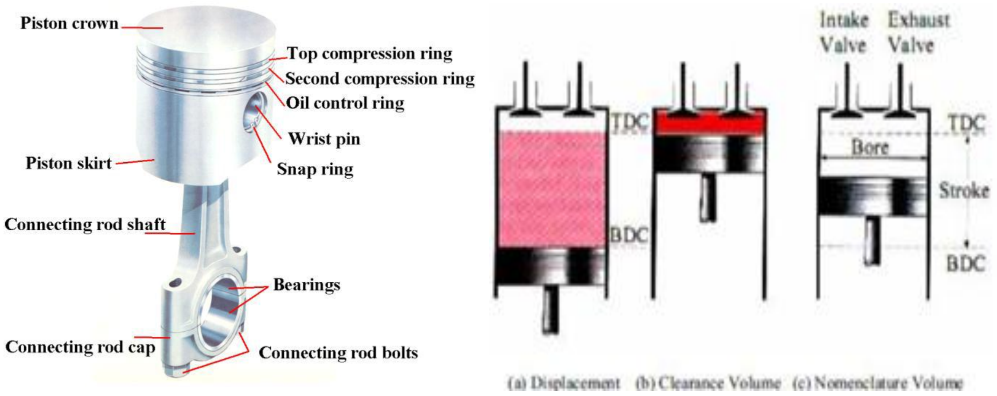

Figure 1.1: Image of a Piston and Con-Rod.

The piston is the heart of the internal combustion engine and is subjected to loads such as thermal and structural stress. The piston reciprocates within the cylinder. The two extremes of this motion are referred to as Top Dead Center (TDC) and Bottom Dead Center (BDC) shown in Fig. 1.2. Top Dead Center is the position of the piston that creates the smallest volume in the cylinder, which is defined as the clearance volume, V_c. This is where combustion takes place in the engine and is also known as the combustion chamber. The Bottom Dead Center is when the piston creates the largest volume in the cylinder [3]. The distance between TDC and BDC is referred to as the stroke, and the volume which the piston displaces during this moment, is called the displacement volume, V_d. The piston is connected to the crankshaft via the connecting rod. The crankshaft converts the linear motion of the piston into rotational motion.
Finite Element Analysis is a simulation technique which evaluates the behavior of components, equipment and structures for various loading conditions including applied forces, pressures and temperatures. Thus, a complex engineering problem with non-standard shape and geometry can be solved using finite element analysis where a closed form solution is not available. The finite element analysis methods result in the stress distribution, displacements and reaction loads at supports for the model. FEA techniques can be used for mesh optimization, design optimization, material weight minimization, and shape optimization. [2]

Aim and objectives:

1. To optimize the piston model.
2. Investigate and analyze the thermal stress distribution of piston.
3. To calculate the equivalent (Von Mises) stresses and total deformation.

Process Methodology:

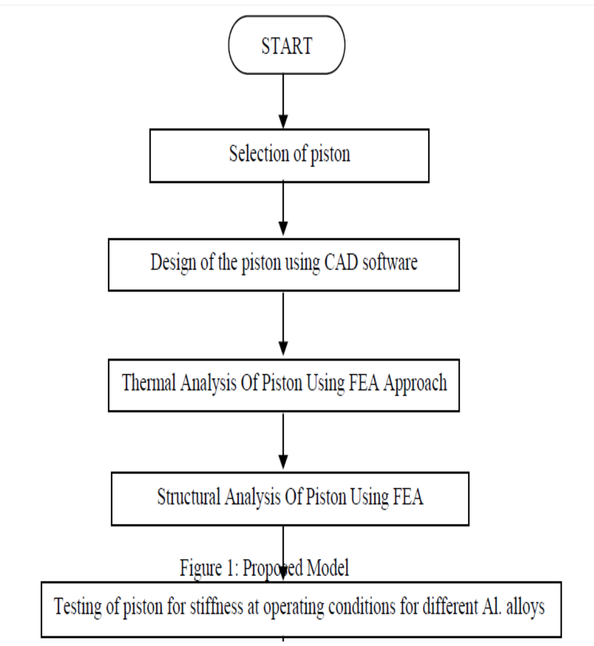

Figure 1: Proposed Model

II. DEVELOPMENT TOOLS

Development Tools:[3]

CATIA (Computer Aided Three-dimensional Interactive Application):

It is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systems. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systems Commonly referred to as 3D Product Lifecycle Management software suite; CATIA supports multiple stages of product development from conceptualization, design (CAD), manufacturing (CAM), and engineering (CAE). CATIA facilitates collaborative engineering across disciplines, including surfacing & shape design, mechanical engineering, equipment and systems engineering.
CATIA started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CAD/CAM CAD software CATIA offers a solution to model complex and intelligent products through the systems engineering approach. It covers the requirements definition, the systems architecture, the behavior modeling.

ANSYS:[3]

The ANSYS Workbench environment is an intuitive up-front finite element analysis tool that is used in Conjunction with CAD systems and/or Design Modeler. ANSYS Workbench is a software environment for performing structural, thermal, and electromagnetic analyses. The class focuses on attaching existing geometry, setting up the finite element model, solving, and reviewing results. The class will describe how to use the code as well as basic finite element simulation concepts and results interpretation. The finite element method (FEM) is a method for dividing up a very complicated problem into small elements that can be solved in relation to each other. Its practical application is often known as finite element analysis (FEA)

ANSYS FEA Procedure: [5]

In general, a finite element solution may be broken into the following three stages.
Preprocessing: defining the problem; the major steps in preprocessing are given below:
Define key points/lines/areas/volumes (Solid Modeling) Define element type and material/geometric properties Mesh lines/areas/volumes as required.

Solution: assigning loads, constraints and solving. Apply the loads (point or pressure), Specify constraints (translational and rotational) Finally solve the problem.

Post processing: further processing and viewing of the results; lists of nodal displacements and show the deformation Element forces and moments Stress/strain contour diagrams.

III. CONCLUSION

From the study it is calculated that the suggested design of piston is more suitable and able to withstand more thermal and structural stresses.

REFERENCES

[1] Silva F.S. Fatigue on engine pistons – A compendium of case studies, Department of Mechanical Engineering, University of Minho, Portugal, Engineering Failure Analysis 13 (2006) 480– 492

[2] Shigley, Joseph E. Mechanical Engineering Design, 3rd edition, 1977 McGraw-Hill.

[3] Jadhav Rajendra B, Dr. Vikhe Patil G. J. Computer Aided Design and Analysis of Piston Mechanism of Four Stroke S.I. Engine, Amrutvahini College of Engineering, Sangamner, Pune University, Maharashtra- 422605; 978-1-4244-9082-0/10 ©2010 IEEE

[4] Richard Stone and Jeffrey K. Ball, Automotive Engineering Fundamentals, SAE International Warrendale. Pa, Copyright © 2004, Richard Stone and Jeffrey K. Ball, ISBN 0-7680-0987-1, SAE Order No. R-199

[5] Shenoy P. S. and Fatemi A. 2006, Dynamic Analysis of Loads and Stresses in Connecting Rods, Journal of Mechanical Engineering Science, Vol. 220, No. 5, pp. 615-624

[6]Swati S Chougule, Vinayak H Khatawat / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, www.ijera.com Vol. 3, Issue 2, March -April 2013, pp.1724-1731